# TransAT

## Release Notes

**Version 5.7.1**

This document summarizes the latest developments brought into TransAT since the previous release.

# Contents

# Chapter 1

# Release 5.7.1: New Features

- Solver
  - Multiphase: enabled VOF and compressibility
  - Compressibility: Peng Robinson Equation of State
  - Phase Change: Antoine saturation curve method
  - Solid Models: Solving for temperature inside solid objects
- User Interface
  - Fast Shrink around Objects
  - Bug Fixes and user experience improvements

## 1.1 Multiphase: enabled VOF and compressibility

The VOF Multiphase feature was added in release 5.7, but was incompatible with the compressible solver. Compressibility has been re-enabled for use with the VOF interface tracking method.

## 1.2 Compressibility: Peng Robinson Equation of State

Previously, Peng Robinson equation of state was available only pre-defined for water-liquid and water-vapor. The equation has been generalized and is available by providing Peng-Robinson parameters. The necessary parameters needed per phase are: Acentric Factor, Volume Translation, Critical Pressure and Critical Temperature.

## 1.3 Phase Change: Antoine saturation curve method

For Phase Change models using Thermally limited phase change and/or Inertial Phase Change/-Cavitation model, the saturation temperature and/or saturation pressure can be defined by the Antoine equation. If defined, the parameters A [bar], B [Kelvin] and C [Kelvin] are required for the relation. Pre-defined saturation curves for phase change of water or phase change of ammonia are still available.

## 1.4 Solid Models: Solving for temperature inside solid objects

Temperature distribution inside solid objects can be solved for in steady simulations. This was introduced for solids with Neumann temperature boundary conditions (specified heat flux).

## 1.5 Fast Shrink around Objects

Shrinking blocks around CAD objects can be done with a Fast method. The Shrink algorithm consists of two parts:

1. Removing blocks that do not intersect the CAD object (inside and/or outside the object)

2. Shrinking blocks that are intersecting the object

The "Fast Shrink" only performs the first step. A "Detailed Shrink" option is available, which includes the second step.

## 1.6 UI: Bug Fixes and improvements

- robust file operations (loading CAD from stl)
- easy reset mesh button (mesh view, upper left)
- easy reset view button (mesh view, lower left)
- optimized CAD graphics: render a coarsened version of the file

# Chapter 2

# Release 5.7

- Solver
  - New 6-DOF rigid body motion and coupled motion module
  - New multiphase viscoelastic rheology module
  - Advanced turbulent heat flux modelling
  - User defined body forces
  - Regularisation of non-Newtonian fluid models with yield
  - User defined variable for Paraview output
- User Interface
  - Full support for new rigid body motion and viscoelastic modules
  - Simplified handling of embedded solids
  - Additional UDF templates available
- Deprecated Features
  - Surface and porous reactions in the Species & Reaction module are no longer available

## 2.1 New 6-DOF rigid body motion and coupled motion module

Embedded solids are now allowed to move freely in three-dimensional space (six degrees of freedom).

From the graphical user interface (GUI) solids can be defined with translational and rotational velocities in three-dimensional space. The user defined functions (UDF) interface was extended to give full control over the object motion, which can be used to define arbitrary trajectories if needed.

Alternatively, the solid motion can be coupled to the fluid motion, i.e. the solid movement is dictated by fluid forces and body forces acting on the solid.

Global forces and torques acting on each solid are output directly in a data table format.

## 2.2 New multiphase viscoelastic rheology module

Simulation of single and two-phase flow of viscoelastic fluids is now available in both single-mode and multi-mode versions. In the multiphase formulation the interface between two immiscible phases are tracked using the level-set method, enabling the simulation of blow molding and extrusion molding processes, as well as covering a multitude of other application areas.

The following models are implemented:

- Upper Convected Maxwell (UCM)
- Oldroyd-B
- Linear Phan-Thien-Tanner (Linear PTT)
- Exponential Phan-Thien-Tanner (Exponential PTT)

The Log Conformation Tensor (LCT) method is available for high Weissenberg number problems.

## 2.3 Advanced turbulent heat flux modelling

Algebraic turbulent heat flux models (ATHFM) are now available for the user in combination with 2-equation turbulence models in various flavours. In addition, a full (partial differential) turbulent heat flux model (THFM) solving for the three components of the turbulent heat flux vector is also available which is augmented by the temperature variance and dissipation of temperature variance equations.

## 2.4 User defined body forces

Addition of user defined function (UDF) interface to allow the application of arbitrary custom body forces using the existing UDF module.

## 2.5 Regularisation of non-Newtonian fluid models with yield

Improved stability of non-Newtonian fluid flow simulations by preventing large values for model viscosity values of the Bingham and Hershel-Bulkley Papanastasiou rheologies which include a yield stress.

## 2.6 User defined variable for Paraview output

Addition of user defined function (UDF) interface that allows the definition of a custom field in the Paraview output. Support for all existing UDF interfaces to a multitude of simulation parameters opens up many possibilities for custom output, rendering and post-processing.

# Chapter 3

# Release 5.6: Changes

- Solver
  - Rosseland Radiation model
  - Periodic Boundary Conditions
  - Multiphase inputs for Pressure Forcing (Periodic BCs)
  - Yap Correction for Turbulent Flows
  - Swirl Correction for Turbulent Flows
  - Dilatation Dissipation for Compressible Turbulent Flows
  - Embedded Fan
  - Fire Source
  - Pressure Loss Coefficient for Outlets
  - Thermal Marangoni effect
- User Interface
  - Paraview Results written as binary
  - Improved exporting of CAD to Mesh
  - Miscellaneous bug fixes and other improvements
  - Extension of the variables available through the C++ interface.

# Chapter 4

# Release 5.5: Changes

- Solver
  - Pressure gradient approximation for algebraic slip model (ASM)
  - Interphase drag, lift, and wall lubrication: set models between each phase combination
  - User Defined Functions for ASM: drag, lift, and wall lubrication models
  - UDF source terms: alpha-rho, mass fraction, pressure, concentration, epsilon, TKE
  - Access to concentration diffusivity in the properties module
  - Compressibility allowed with algebraic slip model
  - Information provided during simulation about ASM validity
  - New phase change model for cavitation
  - Heat capacity calculation from equation of state
- User Interface
  - Momentum transfer tables to define interphase models: drag, lift, and wall lubrication
  - New window for UDFs: writing, compiling and executing UDFs
  - Templates for UDFs
  - Creation of advanced initial conditions in the initial conditions window